FINITE ELEMENT ANALYSIS OF STEEL FRAME HIGH RISE USING ANSYS

YAP KAI XIANG

B. ENG(HONS.) CIVIL ENGINEERING

UNIVERSITI MALAYSIA PAHANG

SUPERVISOR'S DECLARATION

I/We* hereby declare that I/We* have checked this thesis/project* and in my/our* opinion, this thesis/project* is adequate in terms of scope and quality for the award of the Bachelor Degree of Civil Engineering

_____

(Supervisor's Signature)

Full Name :

Position :

Date :

_____

(Co-supervisor's Signature)

Full Name :

Position :

Date :

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is based on my original work except for quotations and citations which have been duly acknowledged. I also declare that it has not been previously or concurrently submitted for any other degree at Universiti Malaysia Pahang or any other institutions.

_____

(Student's Signature)

Full Name : YAP KAI XIANG

ID Number : AA14219

Date :

FINITE ELEMENT ANALYSIS OF STEEL FRAME HIGH RISE USING ANSYS

YAP KAI XIANG

Thesis submitted in fulfillment of the requirements
for the award of the
Bachelor Degree in Civil Engineering

Faculty of Civil Engineering and Earth Resources

UNIVERSITI MALAYSIA PAHANG

JUNE 2018

ACKNOWLEDGEMENTS

First of all, I would like to give thanks to my Supervisor, Dr. Cheng Hock Tian. Along this whole research, he had provided a lot of guidance and helps to me in order to finish my research as well as thesis writing. Besides that, he also provides few examples and tutorials for me to enrich my knowledge regarding the software used.

Furthermore, I truly appreciated the encouragements that have been given by my family members throughout the final year project. Other than that, I would also like to speak a million thanks to my course mates. This is because that they have also been helping out by providing some suggestions and advices, so that I am able to complete my final year project. Without their supports, I am unable to be completing my final year project smoothly.

Moreover, I feel glad that UNIVERSITY MALAYSIA PAHANG has provided me an opportunity to complete my research and final year project. I feel contented as well, because I am able to learn using ANSYS software right before graduation. This is one of the greatest experiences in university life. I am very appreciated for all of it.

ABSTRAK

Kajian ini berkaitan dengan analisis bingkai peningkatan tinggi dengan menggunakan kedua-dua perhitungan manual dan program ANSYS dengan kaedah probabilistik. Dengan menggunakan analisis ini, tingkah laku struktur, seperti tekanan, ketegangan, pesongan dan ubah bentuk boleh ditentukan. Strukturnya akan diperiksa mengikut Eurocode 3 untuk menentukan sama ada tingkah lakunya melewati semua pemeriksaan kod. Bingkai portal biasanya digunakan untuk bangunan perindustrian, tetapi bukan untuk bangunan kediaman. Oleh itu, bingkai portal konkrit bertetulang yang sedia ada telah digunakan sebagai asas untuk pemodelan dan analisis rangka baja portal. Oleh kerana kualiti keluli yang lebih baik dan kos membina keseluruhan yang rendah, keluli dengan 500 N / mm2 kekuatan hasil telah dipilih untuk analisis bingkai portal. Sepanjang analisis, bahan dan geometri yang digunakan untuk struktur telah dipenuhi untuk penyelidikan ini kerana ia melepasi semua pemeriksaan kod. Dari hasil yang diperoleh daripada proses simulasi, kami dapat mengetahui tingkah laku sebenar struktur di bawah parameter input rawak. Selain itu, hasil fungsi pengedaran kumulatif, plot histogram, plot kepekaan dan plot sejarah sampel oleh 10000 kali simulasi diperoleh melalui analisis probabilistic.

ABSTRACT

This research deals with the analysis of a high rise frame by using both manual calculation and ANSYS program with the probabilistic method. By using this analysis, behaviors of the structure, such as stress, strain, deflection and deformation can be determined. The structure will then be checked according to Eurocode 3 to determine whether its behaviors pass all the code checking. Portal frame is commonly used for industrial building, but not for residential building. Thus, an existing reinforced concrete portal frame was used as the basis for the steel portal frame modeling and analysis. Due to the better quality of steel and low overall constructing cost, steel with 500 N/mm^2 of yield strength was chosen for the portal frame analysis. Throughout the analysis, the materials and geometry used for the structure were satisfied for this research since it passed all the code checking. From the results obtained from simulation process, we able to know the real behaviors of the structure under random input parameters. Besides, results of cumulative distribution function, histogram plot, sensitivity plot and sample history plot by 10000 times of simulation were obtained through probabilistic analysis.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

A	*Area of section*
b	*Width*
c	*End Clearance*
d	*Depth*
E	*Modulus of Elasticity*
G	*Modulus of Rigidity*
h	*Height*
I	*Moment of Inertia*
i	*Radius of Gyration*
I_T	*Torsional Constant*
I_W	*Warping Constant*
K	*Global-Coordinate Structure Stiffness Matrix*
L_{cr}	*Buckling Length*
M	*Moment*
N	*Axial Load*
r	*Root Radius*
tf	*Flange Thickness*
tw	*Web Thickness*
W_{pl}	*Plastic Modulus*
X	*Reduction Factor*
α	*Imperfection Factor*
γ_{M0}	*Partial Factor for Resistance of Cross-Sections Whatever The Class Is*
γ_{M1}	*Partial Factor for Resistance of Members to Instability Assessed by Member Checks*
λ	*Slenderness Value Slenderness Value*
Φ	*Value to Determine The Reduction Factor*

LIST OF ABBREVIATIONS

2D	*2 Dimensional*
3D	*3 Dimensional*
BMSHPRO	*Beam and Shell Properties*
CBFEM	*Composite-Based Finite Element Model*
CDF	*Cumulative Distribution Function*
CFD	*Computational Fluid Dynamics*
ChckAxis	*Check Axis*
CIVILFEM	*Civil Finite Element Method*
CS	*Coordinate System*
DMAX	*Maximum Displacement*
DOE	*Design of Experiment*
DOF	*Degree of Freedom*
ELASTIC	*Modulus of Elasticity*
FE	*Finite Element*
FEM	*Finite Element Model*
GAUS	*Gaussian*
Inp	*Input*
Ku	*Kurtosis*
LatBuck	*Lateral Buckling*
LS	*Load Step*
Max	*Maximum*
Min	*Minimum*
Out	*Output*
P1	*Vertical Load*
P2	*Wind Load*
Par1	*Mean*
Par2	*Standard Deviation*
PDF	*Probabilistic Density Function*
POISSON	*Poisson ratio*
PRES	*Pressure*
RSM	*Response Surface Methodology*

S.D.	*Standard Deviation*
Sk	*Skewness*
TEMP	*Temperature*
UB	*Universal Beam*
UC	*Universal Column*
UDL	*Uniform Distributed Load*

CHAPTER 1

INTRODUCTION

1.1 Introduction

Framing is a very basic structure in any constructions. It is made up of several beams and columns to provide supports and shape to the structures, which its materials are generally made of wood, engineered wood, or structural steel. Nowadays, steel has become an important material in construction companies, as it has more advantages than timbers and other materials. Because of this, steel frame becomes a very common form of building in other countries like New Zealand and Australia (Moss et al., 2009), however, it is still not yet fully applied for framing in Malaysia, as there are concerns regarding the probable risks on steel framing system (Madsen, 2005).

There are several types of frame available in steel construction industries. Among them, portal frame is the most common and widely used especially for single-story industrial building. The usage of steel frames by non-domestic buildings is more than 90% has proven that the dominance of steel construction these buildings (Davidson & W. Owens, 2012). The properties of steel such as light, long-span and high durability have allowed them to be chosen as the material for portal frames. Besides that, comparing with concrete, steel is easier to be obtained and erected quickly. These enable architects to design more economical and attractive in appearance buildings. Portal frame has better stability against lateral and vertical actions, and these are the reasons portal frame is the best choice for residential buildings. As portal frame can be very rigid and stable in plane with moment-resisting connections (Davidson & W. Owens, 2012).

The 3D portal frame that will be used for analysing in this research is a high rise building with 49 m in height, 80 m in length and 32 m in width. Figure 1.1 shows the detailed dimensions of this high rise building. Since it is a high rise building, it will be

sustaining wind load at both service limit state and ultimate limit state, therefore, wind load is considered in this research.

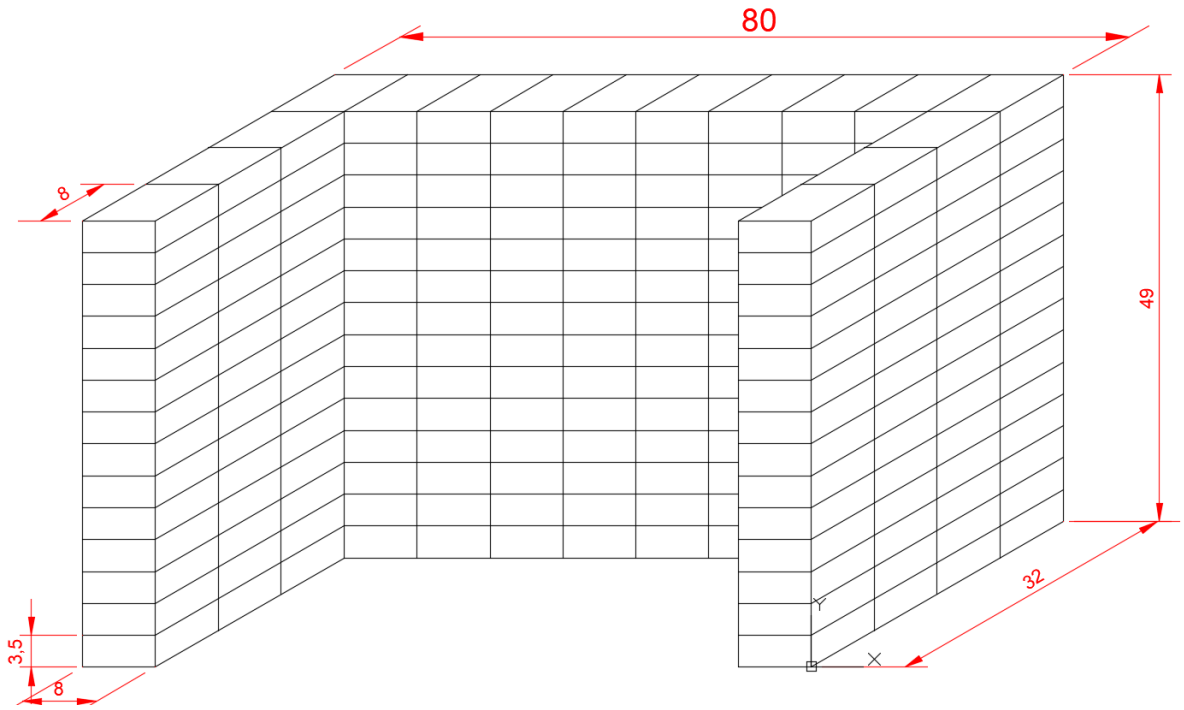

Figure 1.1: Dimensions of High Rise Building in Detail

1.2 Problem Statement

2D Portal frame is easier to be analysed manually. However, a 3D portal frame is a very complicated structure, it consumes a lot of time to analyse its behaviour and cause higher cost in designing the 3D portal frame. There are a lot of software can be used for analysing portal frame, and ANSYS is one of them that very is useful in assisting to analyse complicated structures, as well as easy to model a 3D structure using it. By using ANSYS, it helps to save a lot of time and cost when designing a structure.

Therefore, ANSYS + CIVILFEM 12.0 program will be used for modelling and stimulating the characteristic behaviours of the steel frame in this research. CivilFEM, is a civil engineering special software that comes in package with ANSYS, is taking base on the structure of civil engineering for a variety of simulation of design and checking (Moreno et al., 2001). During this research, maximum deflection, deformed shape, and variety of checking will be done according to Eurocode 3.

1.3 Objective

In order to well conduct researches, objectives are important part for every project or thesis, and they are needed to be set well and logically. It is the guidance for researchers to achieve the final objective and avoid researches to digress. Objectives also assist the researchers to understand and keep reminding themselves of the tasks that should be done in the research. Therefore, the main objectives of this research are:

(i) To determine the structural behaviours of 3D portal frame
(ii) To ensure the portal frame passes all the code checking
(iii) To make sure the portal frame is stable against deflection and deformation
(iv) To determine the response behaviours of portal frame under deflection
(v) To determine the behaviours of portal frame under surface load and wind load

1.4 Scope of Study

In this research, ANSYS + CIVILFEM 12.0 will be used to carry out 3D portal frame modelling and analyzation. The loading that acts on every beams were calculated according to Eurocode 1, which includes the weights of slabs and brick walls, and will be applied as pressure on beam in CivilFEM. Meanwhile, wind load is calculated according to Malaysia Standard, and it will be applied on only one side of the structure. Lastly, deflection, deformed shape and Eurocode 3 checking will be done in post-processing step.

1.5 Expected Outcome

This research claims to find out the structural behaviours and whether it will pass the code checking of 3D portal frame. The behaviours are deformation, deflection, tension checking, compression checking and lateral torsional buckling checking according to Eurocode 3.

REFERENCES

-, J. J., -, F. X., -, W. Z., -, D. X., & -, Q. D. (2012). Static Performance Analysis of Large Span Portal Frame with Variable Section. *International Journal of Digital Content Technology and Its Applications*, *6*(12), 73–82. https://doi.org/10.4156/jdcta.vol6.issue12.9

ANSYS. (January, 2018). *4.188 BEAM188 3-D Linear Finite Strain Beam*. Retrieved from ANSYS: http://www.ansys.stuba.sk/html/elem_55/chapter4/ES4-188.htm

BS EN 1991-1-1:. (2002). *Eurocode 1: Actions on structures. General actions. Densities, self-weight, imposed loads for buildings , BSI.*

Caprani, C. (2010). Plastic Analysis 3rd Year Structural Engineering, (January), 1–129.

Carley, K. M., Kamneva, N. Y., & Reminga, J. (2004). Response surface methodology. *CASOS Technical Report*, (October), 1–26. https://doi.org/10.1002/wics.73

Davidson, B., & W. Owens, G. (2012). *STEEL DESIGNERS' MANUAL*. BLACKWELL PUBLISHING LTD.

Ding, Y., Song, X., & Zhu, H. (2017). Probabilistic progressive collapse analysis of steel frame structures against blast loads. *Engineering Structures*, *147*, 679–691. https://doi.org/10.1016/j.engstruct.2017.05.063

Duoc, T., James B.P., L., Tiku T. , T., R. Mark, L., Yixiang, X., Steven, M., & Wei, S. (2013). Effect of serviceability limits on optimal design of steel portal frames. *Journal of Constructional Steel Research*, 74-84.

El-Heweity, M. M. (2012). Behavior of portal frames of steel hollow sections exposed to fire. *Alexandria Engineering Journal*, *51*(2), 95–107. https://doi.org/10.1016/j.aej.2012.06.004

Elsayed, M., Mohamed, E.-H., Hamdy, A.-E., & Mohamed, O. (2010). Finite element analysis of beam-to-column joints. *Alexandria Engineering Journal*, 91-104.

Fu, F. (2009). Progressive collapse analysis of high-rise building with 3-D finite element modeling method. *Journal of Constructional Steel Research*, *65*(6), 1269–1278. https://doi.org/10.1016/j.jcsr.2009.02.001

G Lackshmi, N. (2009). *Finite Element Analysis.* BS Publications.

Goswami, S., Ghosh, S., & Chakraborty, S. (2016). Reliability analysis of structures by iterative improved response surface method. *Structural Safety*, *60*, 56–66. https://doi.org/10.1016/j.strusafe.2016.02.002

Gunst, R. F. (1996). Response Surface Methodology: Process and Product Optimization Using Designed Experiments. *Technometrics*, *38*(3), 284–286. https://doi.org/10.1080/00401706.1996.10484509

Iu, C. K., Chen, W. F., Chan, S. L., & Ma, T. W. (2008). Direct second-order elastic analysis for steel frame design. *KSCE Journal of Civil Engineering*, *12*(6), 379–389. https://doi.org/10.1007/s12205-008-0379-3

Lausova, L., Kolos, I., Michalcova, V., & Skotnicova, I. (2017). Numerical Analysis of Steel Portal Frame Exposed to Fire. *Procedia Engineering*, *190*, 237–242. https://doi.org/10.1016/j.proeng.2017.05.332

Lubomír Šabatka, František Wald, Jaromír Kabeláč, Drahoslav Kolaja, & Martin Pospíšil. (2015). Structural Analysis and Design of Steel Connections Using Component-Based Finite Element Model. *Journal of Civil Engineering and Architecture*, *9*(8), 895–901. https://doi.org/10.17265/1934-7359/2015.08.002

Lujan-moreno, G. A., Howard, P. R., Rojas, O. G., & Montgomery, D. C. (2018). PT US CR. *Expert Systems With Applications*. https://doi.org/10.1016/j.eswa.2018.05.024

M.T., R.-L., & Jose, S.-S. (2014). Analysis of wind action on unique structures with application to Seville. *Engineering Structure*.

Madsen, J. J. (6 January, 2005). *Which is the better building material? Concrete or Steel?* Retrieved from Buildings Smarter Facility Management: https://www.buildings.com/article-details/articleid/2511/title/which-is-the-better-building-material-concrete-or-steel-/viewall/true

Moreno, M. Á., Monteagudo, E., Maia, I., & Ingeciber, S. A. (2001). ANSYS + CivilFEM : H igh -E nd S olution F or A dvanced C ivil Engineering Projects.

Moss, P. J., Dhakal, R. P., Bong, M. W., & Buchanan, A. H. (2009). Design of steel portal frame buildings for fire safety. *Journal of Constructional Steel Research*, *65*(5), 1216–1224. https://doi.org/10.1016/j.jcsr.2008.09.003

P.J., M., R.P., D., M.W. , B., & A.H. , B. (2008). Design of steel portal frame buildings for fire safety. *Journal of Constructional Steel Research*, 1216-1224.

Phan, D. T., Lim, J. B. P., Tanyimboh, T. T., Lawson, R. M., Xu, Y., Martin, S., & Sha, W. (2013). Effect of serviceability limits on optimal design of steel portal frames. *Journal of Constructional Steel Research*, *86*, 74–84. https://doi.org/10.1016/j.jcsr.2013.03.001

Reh, S., Beley, J.-D., Mukherjee, S., & Khor, E. H. (2006). Probabilistic finite element analysis using ANSYS. *Structural Safety*, *28*(1–2), 17–43. https://doi.org/10.1016/j.strusafe.2005.03.010

Ross, M., James, B., Tiku, T., Duoc, T., & Wei Sha. (2014). Optimal design of long-span steel portal frames using fabricated beams. *Journal of Constructional Steel Research*, 104-114.

SHARCNet. (January, 2018). *BEAM 188*. Retrieved from SHARCNet: https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/ans_elem/Hlp_E_BEAM188.html

Steel Construction. (2014). Retrieved from Steel Construction: http://www.steelconstruction.info/Portal_frames

University of Alberta - ANSYS Tutorials. (2001). Retrieved from http://www.mece.ualberta.ca/tutorials/ansys/